Efficient solving strategies for incompressible Navier-Stokes equations for large scale simulations using the open source software Feel++

Vincent Chabannes, Christophe Prud'Homme, Marcela Szopos, Ranine Tarabay

Efficient solving strategies for incompressible Navier-Stokes equations for large scale simulations using the open source software Feel++

V. Chabannes*, C. Prud'homme, M. Szopos** and R. TARABAY****

* Laboratoire Jean Kuntzmann, Université Joseph Fourier, France

** Institut de Recherche Mathématique Avancée, Université de Strasbourg, France

Keywords: Navier-Stokes equations, large scale simulations, computational fluid dynamics, scalable parallel preconditioners, experimental validation, Feel++

Over the past few decades, the computational fluid dynamics has evolved to become an important tool for the description of the complex multi-physics, multi-scale phenomena characterizing blood flow. Its reliability depends both on the verification of the numerical methods and on the validation of the mathematical models.

The aim of the first part of the talk is to present a preconditioning framework for the linear system arising from the finite element discretizations and time advancing finite difference schemes of the 3D steady and unsteady Navier-Stokes equations. In particular, we are interested in preconditioners based on an algebraic factorization of the system's matrix which exploit its block structure, such as the Pressure Convection-Diffusion (PCD) preconditioner , the SIMPLE preconditioner or the LSC preconditioner, see [Elman et al. (2014)]. A comparison between the efficiency of these preconditioners is ascertain by testing them over the 3D backward facing step benchmark. The iteration counts using the PCD preconditioner are independent of mesh size and high order finite elements and mildly dependent on Reynolds numbers for steady flow problems which is not the case for the other preconditioners.

In the second part of the talk we describe a framework for the solution of flow problems relevant to biomechanics strongly supported by the aforementioned solving strategies. We assess the efficiency of this framework through experimental data for fluid flow in a nozzle model with rigid boundaries, a device designed to reproduce acceleration, deceleration and recirculation, features commonly encountered in medical devices [Stewart et al. (2012)]. The flow rates were chosen to cover laminar ($Re = 500$), transient ($Re = 2000$) and turbulent ($Re = 3500$) regimes.

The numerical results displayed during the presentation are obtained using the open-source library Feel++ (Finite Element method Embedded Language in C++, www.feelpp.org).

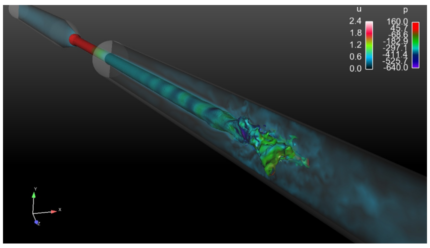

Figure 1: Computational Fluid Dynamics FDA Benchmark at $Re = 2000$.

	Re 10		Re 200		Re 400	
h	P2P1	P3P2	P2P1	P3P2	P2P1	P3P2
0.125	22[4]	20[4]	17[11]	15[11]	20[14]	16[15]
0.09375	21[4]	18[4]	17[11]	14[11]	18[14]	15[15]
0.0625	18[4]	17[4]	14[12]	12[11]	16[14]	14[15]

Table 1: Number of outer iterations (using PCD) at the last nonlinear iteration of the Picard algorithm and [the total number of nonlinear iterations] for the 3D step geometry

References

Elman, H. C., D. Silvester, and A. Wathen (2014). *Finite elements and fast iterative solvers: with applications in incompressible fluid dynamics*. Oxford University Press.

Stewart, S., E. Paterson, G. Burgreen, P. Hariharan, M. Giarra, V. Reddy, S. Day, K. Manning, S. Deutsch, M. Berman, M. Myers, and R. Malinauskas (2012). Assessment of cfd performance in simulations of an idealized medical device: Results of fda's first computational interlaboratory study. *Cardiovascular Engineering and Technology 3*(2), 139–160.

Tarabay, Efficient solving strategies for incompressible Navier-Stokes equations for large scale simulations using the open source software Feel++, 7 rue René Descartes 67084 Strasbourg, 0368850232, rtarabay@math.unistra.fr